Title: Block-Coupled Simulations Using OpenFOAM

Instructor: Ivor D. Clifford, Penn State University

Description: In this class, the recently-developed block-coupled solver for OpenFOAM is introduced. The basic approach, theory, applicability and limitations of the solver are discussed. We start with a simple two-phase coupled diffusion problem and demonstrate how the block-coupled solver may be applied to achieve stable, tightly converged solutions to an inherently stiff set of equations. Issues such as multi-variable coupling, multi-region problems, and parallel processing support will also be addressed.

Duration: 90 minutes

Date and Location: 13 June 2011, Penn State University, State College, PA, USA